TENTH AUSTRALASIAN FLUID MECHANICS
CONFERENCE - UNIVERSITY OF MELBOURNE
11-15 DECEMBER 1989

15A-3

FLOW MEASUREMENT AND NUMERICAL PREDICTION IN AN ISOTHERMAL MODEL FURNACE

Harley MACKENZIE, Robert MIERISCH and John PERRY

Department of Mechanical Engineering
Swinburne Institute of Technology
P.O. Box 218, Hawthorn, Vic. 3122
AUSTRALIA

ABSTRACT

This paper outlines comparison between three dimensional numerical prediction using INFERNO and matched grid experimental results for a generalized model of a tangentially fired furnace with wall mounted slot burners.

INTRODUCTION

Combustion of brown coal, which is 55% to 67% moisture, has presented unique difficulties. Plant capital cost is much higher than for black coal firing and supplementary fuel is required to achieve combustion stability at reduced firing rates. To improve understanding of the relevant combustion and heat transfer processes in brown coal furnaces the Swinburne Institute of Technology has been involved in a collaborative research program with the State Electricity Commission of Victoria (SECV) since 1981.

Comparison of numerical predictions of isothermal flow in a model furnace with experimental results is a critical step toward numerical modelling of combusting flows. The numerical modelling code INFERNO (Mackenzie and Perry, 1988) based on TEACH (Gosman and Ideriah, 1976) has been used to predict mean velocities in three dimensions in a water filled generalized model furnace for a number of flow conditions.

In this paper we present a brief description of the experimental and numerical modelling, the results for symmetrical flow conditions and a brief discussion of the results.

THE EXPERIMENTAL MODEL

Modern boiler plant operated by the SECV for electric power generation uses furnace designs based on pulverised brown coal, slot burner tangentially fired systems. The furnaces are usually a membrane water wall of rectangular cross section (figure 1 and 2) with eight wall mounted burner sets; each serviced by a mill and fuel conditioning system. To pre-dry the fuel approximately 40% of the furnace gas is extracted above the burners and mixed with fuel in the milling system. The fuel mixture is subsequently separated into fuel solids rich and fuel vapour rich streams which feed the lower (main) and higher (vapour) burners respectively.

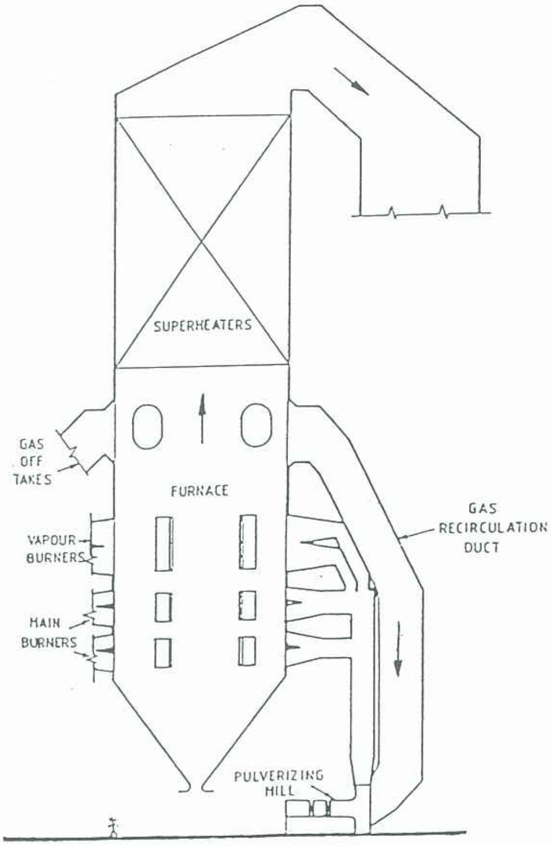


Figure 1. Schematic Cross Section of a Typical Furnace

Each burner consists of three parallel horizontal rectangular ducts above one another and terminating at the furnace wall (figure 3). The centre duct (primary jet) delivers the pre-dried pulverised fuel and resultant carrier gas and the two outer ducts (secondary jets) deliver pre-heated air. The three jets mix as they move into the furnace, entrain hot combustion gases from the furnace which, when conditions are correct, initiate combustion in the furnace flow field.

The design of the experimental model was specifically directed toward providing an improved understanding of the flow processes in the burner/burner and burner/furnace interaction region. Experimental data derived from the modelling programme must be of a form suitable for relevant isothermal validation studies of three-dimensional furnace numerical models.

The general dimensions of the model (figure 2) followed a 1/25th geometric scale of the Loy Yang A furnace. This criterion was set to complement the 1/25th fully geometric scale model of the Loy Yang A furnace located at the SECV and to enable the interchange of modules such as the hopper region and a burner set.

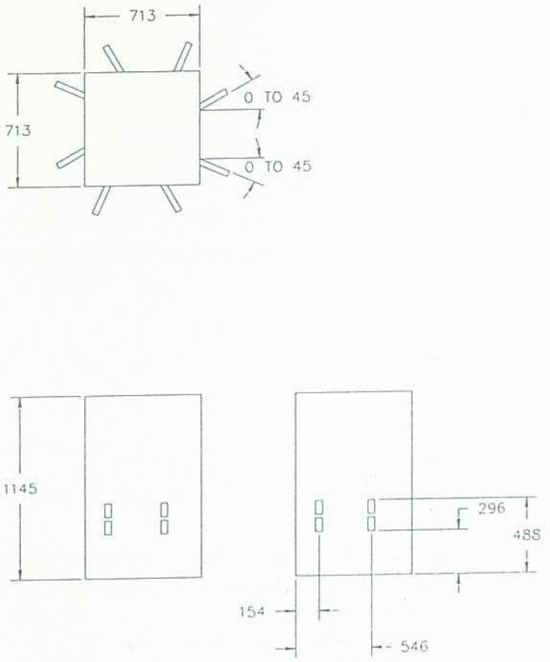


Figure 2. Burner Layout - Model Furnace.

The model was constructed in 300mm high modules with minimum frame thickness to maximise lighting and visibility. The modules are self supporting and the model is mounted approximately 900mm above the laboratory floor on a separate support frame under which all the plumbing is arranged. The space beneath can be used either to locate a large angled mirror to enable the horizontal plane in the model to be video recorded from the side, or to locate lens and mirror units for LDA measurements of horizontal components of flow velocity. The burner and it's recess was modelled as one plain slot (figure 3), with a supply duct of the same cross-sectional dimensions. The resulting slot was 900mm long, with a perforated plate at the upstream end to engender mixing. This ensured near fully developed flow at the burner outlet plane. A water flow of 1.24 l/s (±0.5%) was supplied to each of eight inlet ducts in one plane.

The furnace flow was studied by injecting water mixed with 4 to 6 parts per thousand of 0.55 micrometre monodisperse particles of Titanium Dioxide. Injection points were chosen to match the centre of the grid pattern to be used for the numerical model. Approximately 0.6 ml was injected in 2 seconds, through a 1.3mm diameter stainless steel tube, for each point.

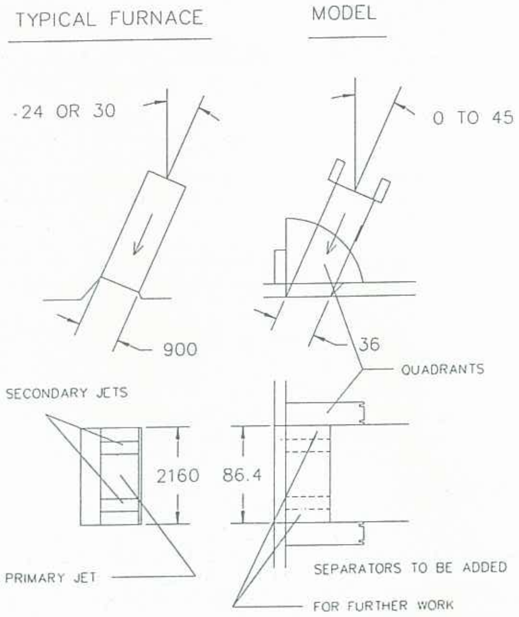


Figure 3. Burner Recess Modelling.

To usefully interpret the tracer pattern it is necessary to simultaneously view the horizontal and one vertical plane. To view and record these observations a split image video system was developed which records the two views simultaneously. Flow patterns have been recorded on video with light levels sufficent to enable a 1/500th second shutter speed to be used resulting in an estimated 0.4mm image blurring. For each point at which pigment was injected the flow direction was estimated and stored in a computer based spreadsheet. This data was sorted and used as an AUTOCAD input file to create streamline diagrams.

THE NUMERICAL MODEL

The present version of the code INFERNO has been developed for isothermal, incompressible, steady state flow using finite difference equations and the solution scheme outlined in Patankar (1985). The unique feature ofINFERNO is the generalised nature of the code, enabling the simple modelling of a wide variety of furnaces as well as the ability to model applications in other unrelated areas such as wind flow over structures and flow in air conditioning systems (Mackenzie and Perry, 1988). The governing time-averaged constant density differential transport equations in cartesian coordinates are the continuity equation, the momentum equation (of which the x-component only is given), and a general transport equation for any variable Φ (Robinson, 1985), as follows:

$$\nabla . V = 0$$

$$\nabla . (\rho V_x V - \mu \nabla V_x) = -\frac{\partial P}{\partial x} + \frac{\partial}{\partial x}\left[\mu \frac{\partial V_x}{\partial x} \right] + \frac{\partial}{\partial y}\left[\mu \frac{\partial V_y}{\partial x} \right] + \frac{\partial}{\partial z}\left[\mu \frac{\partial V_z}{\partial x} \right]$$

$$\nabla . (\rho V \phi - \Gamma_{\phi} \nabla \phi) = S_{\phi}$$

As there is no explicit equation for the pressure, the iterative solution scheme SIMPLE is used to derive implicitly the three velocity components and the pressure field (Patankar, 1985).

One of the more advanced schemes such as the pressure implicit operator splitting PISO technique as described in Issa (1982) was not used because the storage requirements for this approach become prohibitive for three dimensional problems.

The fine scale fluctuations in fluid properties are well beyond resolution by current numerical modelling techniques and indeed are not needed in an engineering simulatiopn, and the turbulent characteristics of the flow must, therefore, be approximated with time averaged quantities rather than instantaneous. The turbulence closure problem is dealt with by the use of the two equationk-ϵ model where k is the turbulent kinetic energy and ϵ the kinetic energy dissipation rate, each of these quantities having its own transport equation and source terms (Gosman and Ideriah, 1976). These turbulent quantities interact with the principal fluid transport equations through the Boussinesq eddy viscosity:

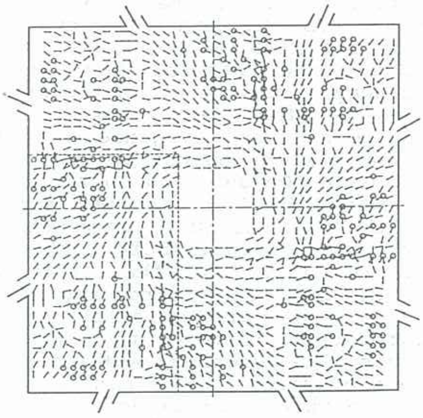
$$\mu_t = C_\mu \rho \kappa^2/\epsilon$$

The k-ϵ model is widely used for internal flows and provides an efficient method for calculating engineering flows, however its performance has been noted to degrade in attached, recirculating, swirl and combusting flows and in high streamline curvature flows (Nallasamy, 1987), and this must be kept in mind in interpreting some features of the solution.

False diffusion has been shown to be of significance in flows inclined to finite difference grid system (Leschziner, 1980) when using the standard hybrid differencing scheme for the advective terms. Various higher order differencing schemes have been developed such as QUICK (Leonard, 1979) and subsequent modifications with improved stability (Freitas et al, 1985). The implementation of these higher order schemes in general three dimensional codes is not straight forward as boundary conditions require artificial property gradients and questions of flow stability arise in the presence of strong source terms as in combusting flows. For simplicity we have chosen to use the efficient Thomas algorithm for the solution of the tridiagonal system (Gosman and Ideriah, 1976) as each control volume is related only to its immediate neighbour cells.

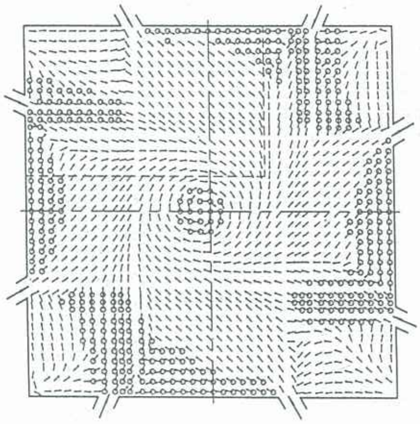
RESULTS

Observations were made over a quarter segment of a horizontal plane just below the top of the burner and a horizontal plane located 120mm above the top of the burner. The numerical model was executed on an IBM3090 computer and yielded velocity data for all grid reference points which was inputed to a streakline post processing routine (Mackenzie and Perry, 1988) to create the flow diagrams. On the diagrams the vertical component is shown as a plain arrow for horizontal or upward flow and an arrow with a circle for downward.



Physical Results

Horizontal plane 7mm below the top of the burner



Numerical Model Results.

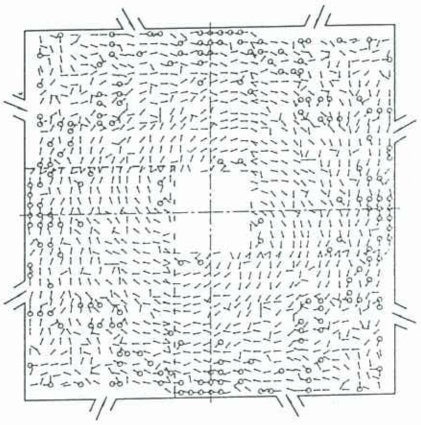
Figure 4. Comparison Between the Observed Flow Pattern and the Numerical Model.

DISCUSSION

Care should be taken in interpreting these results as specific details on flow direction have been derived by viewing the tracer fluid from injection over a finite length which may remain in the plane of interest or divert to a short distance above or below. In many areas the flow was unsteady and accurate inference of a local flow direction difficult. To obtain an appreciation of the accuracy of the method the region of observation was made somewhat greater than a quarter segment (see dashed boundaries on figures 4 and 5) so assuming symmetry about the vertical axis, the observations in the overlap region should match. With few exceptions the direction of the vertical components match and the direction of the horizontal components correspond within 60°. The flow observations were interpreted and recorded prior to the computing of the numerical model. The numerical model solution is not fully converged due to the computer time limitations but convergence is close.

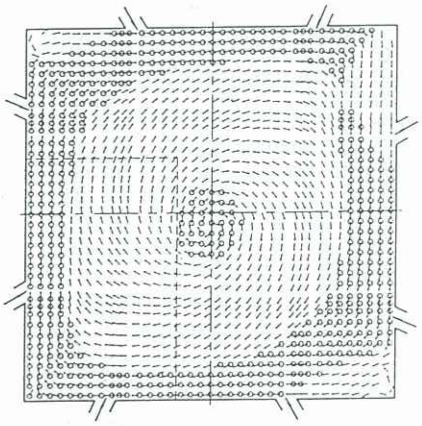
For the burner plane (figure 4) the overall flow pattern corresponds well for the two sources. In both cases the right hand (looking from above) wall jet is diverted, by the rotating flow, to the left and away from the geometric axis of the burner. The left hand burner undergoes a similar divegence but at a greater angle with respect to the burner geometric axis. On one hand in the in the region bounded by the wall and the two wall jets the vertical component of velocity is predominantly downward, particularly close to the left hand burner. On the other hand the corner region has a predominantly upward component except close to the same left hand burner. The numerical model suggests that the flow field in the centre of the model has relatively small downward which has yet to be explored with the hydraulic model.

For the horizontal plane immediately above the burners (figure 5) the flow over the field is clockwise without any apparent sign of the burner jets. There is a suggestion from both sources of a small region of upward moving rotating flow in the corners. The vertical component is small but predominantly down close to the walls, changing to an upward flow toward the central region. The flow suggested by the numerical model has a downward component in the centre.



Physical Results

Horizontal plane 120 mm above the top of the burner



Numerical Model Results.

Figure 5. Comparison Between the Observed Flow Pattern and the Numerical Model.

CONCLUSIONS

Close correspondence of the gross flow characteristics was demonstrated between the numerical and physical model. The flow was unsteady in the region between converging jets in both the wall and the corner regions which increases uncertainty in the accuracy of the flow observations. Accounting for this the level of the correlation for the vertical and horizontal velocity component was encouraging for this symmetrical flow case.

ACKNOWLEDGEMENTS

The authors wish to acknowledge the support of the State Electricity Commission of Victoria , the National Energy Research, Development and Demonstration Council and the Victorian Post Secondary Education Commission for their support in funding this research program. Also the valuable assistance of Prof.Bruce Morton, Graeme Pleasance ,Dr Fred Lockwood, Dr Leigh Peterson, Mr Rudy Gris and the laboratory staff in Mechanical Engineering at Swinburne Institute of Technology is gratefully appreciated.

BIBLIOGRAPHY

Freitas C J, Street R L, Findikakis A N, and Koseff J R, Numerical Simulation of Three Dimensional Flow in a Cavity, International Journal for Numerical Methods in Fluids, Vol 5, 1985, pp. 561-575.

Gosman A D and Ideriah F J K, TEACH - T: A General Program for Two Dimensional Recirculating Flows, Imperial College Fluids Section Report, 1976

Issa R I, Solution of the Implicitly Discretised Fluid Flow Equations by Operator Splitting, Imperial College Fluids Section Report, FS/82/15,September 1982.

Leonard B P, A Stable and Accurate Convective Modelling Procedure Based on Quadratic Upstream Interpolation, Computer Methods in Applied Mechanics and Engineering, Vol 19, 1979, pp.59-98.

Leschiner M A, Practical Evaluation of Three Finite Difference Schemes for the Computation of Steady State Recirculating Flows, Computer Methods in Applied Mechanics and Engineering, Vol 23, 1980, pp.293-312.

Mackenzie H and Perry J H, The Numerical Modelling of the Interaction of Burner Jets in Brown Coal Fired Boilers, Final report for NERDDP Grant 85/5002. 1988

Nallasamy M, Turbulence Models and their Application to the Prediction of Internal Flows, Computers in Fluids, Vol 15, No. 2 1987, pp. 151-194.

Patankar S V, Numerical Heat Transfer and Fluid Flow (Hemisphere Publishing Corporation), 1985

Robinson G F, A Three Dimensional Analytical Model of a Large Tangentially Fired Furnace, Journal of the Institute of Energy, September 1985, pp. 116-150.